

CFD ANALYSIS TO UNDERSTAND THE FLOW BEHAVIOUR OF A SINGLE STAGE TRANSONIC AXIAL FLOW COMPRESSOR

M T Shobhavathy
Scientist, Propulsion Division, CSIR- NAL
Bangalore, India. 560037
Email : mtshobha@nal.res.in

Premakara Hanoca M.Tech Project Trainee,
Propulsion Division,
CSIR- NAL Bangalore, India. 560037
Email: hanocap_87@yahoo.co.in

ABSTRACT

This paper comprises the Computational Fluid Dynamic (CFD) analysis to investigate the flow behaviour of a high speed single stage transonic axial flow compressor. Steady state analyses were carried out at design and part speed conditions to obtain the overall performance map using commercial CFD software ANSYS FLUENT. Radial distribution of flow parameters were obtained at 90% of design speed for the choked flow and near stall flow conditions. The predicted data were validated against available experimental results. The end wall flow fields were studied with the help of velocity vector plots and Mach number contours at peak efficiency and near stall flow conditions at 60% and 100% design speeds. This study exhibited the nature of a transonic compressor, having strong interaction between the rotor passage shock and the tip leakage vortex at design speed, which generates a region of high blockage in the rotor blade passage. The influence of this interaction extends around 15% of the blade outer span at design speed and in the absence of blade passage shock at 60% design speed, the influence of tip leakage flow observed was around 8%.

NOMENCLATURE

P_0	Total pressure (Pa)
p_s	Static pressure (Pa)
M_{rel}	Relative Mach number
D	Diameter(m)
D_t	Tip diameter
D_h	Hub diameter
R	Local radius(m)
R_t	Tip radius
R_h	Hub radius
RPM	Revolutions per minute

INTRODUCTION

Transonic axial flow compressors are widely used in aircraft engines to obtain high pressure ratios per stage. High stage pressure ratios are important as they make it possible to reduce the engine weight and size and, thus, investment and operational costs [1]. Performance of transonic compressors has reached a high level but engine manufacturers are oriented towards increasing it further. The flow field inside a transonic compressor rotor is extremely complex and presents many challenges to compressor designers, who have to deal with several and concurring flow features such as shock waves, intense secondary flows, shock/boundary layer interaction, etc., inducing energy losses [2]. Interacting with secondary flows, shock waves concur with development of blockage [3]. Predominantly detrimental is the interaction with tip clearance flow at the outer span of the rotor, where the compressor generally shows the higher entropy production [4, 5 and 6].

For designers and researchers, prediction of operating characteristics performance of turbomachinery is most important task. All theoretical methods for predicting the performance merely gives a value, and one is unable to determine the root cause for the poor performance. Though experimental method is the ultimate scheme, it has its own disadvantages. Nowadays, much progress has been made in the field of Computational Fluid Dynamics (CFD) and is therefore being increasingly used in industry by designers and researchers to optimize aerodynamic designs, for predicting performance at various operating conditions to aid understanding of fluid flow phenomena and consecutively complementary to experimental testing.

Adamczyk et al [7] and Copenhaver et al [8] have used 3D Navier-Stokes solvers to study the shock /vortex interaction. Adamczyk studied the effect of variations in tip clearance on the performance of a transonic rotor, and showed that the shock/vortex interaction plays a major role in determining the compressor flow range. He presented how different mathematical models can be used to simulate turbomachinery flows and particularly how the 3-Dimensional

(3D) calculation can help the multistage turbomachinery development activity. Storer and Cumpsty[9] worked on tip clearance loss in axial flow compressors and have shown that the losses due to tip leakage flow are primarily associated with mixing process that takes place between the leakage flow and the through flow, and these losses can be predicted from the angle formed between the clearance flow and the through flow. Based on the work of Koch and Smith [10], it is clear that there is a relationship between the blockage in a turbomachine and the losses, pressure rise, and flow range of that turbomachine. Smith demonstrated that for low speed axial compressors the end wall boundary layer thickness is directly related to: i) the blade-to-blade passage width, ii) the aerodynamic loading level and iii) the tip clearance. A methodology to quantify the end wall blockage generated within the blade row by the tip clearance flow was developed by Khalid [11]. His results were based on 3-Dimensional Navier- Stokes computations of the flowfields in a low speed stator, low speed rotor, and a transonic fan with several values of tip clearance height. His results indicated that the loss in total pressure in the end wall region resulted from the interaction of the leakage flow and passage flows and the vortical structure associated with the clearance vortex was not a major factor in generating the end wall blockage. K.L. Sudar et al [3] have implemented both experimental and computational techniques to investigate the tip clearance flows in a transonic axial compressor rotor at design and part speed conditions. They focused particularly on the role of passage shock/leakage vortex interaction in generating end wall blockage. They found that the strong interaction between the rotor passage shock and tip leakage vortex generates a region of high blockage in the passage which moves forward and becomes larger in both the circumferential and radial directions as the loading is increased.

This paper describes the application of commercial 3-D CFD code ANSYS FLUENT to investigate the flow behaviour of a high speed single stage transonic axial flow compressor.

The main objective of this study is to benchmark the CFD code and hence it could be used further for the parametric studies on transonic compressor as a complementary to the experimental work carried out on the research compressor at CSIR-National Aerospace Laboratories.

The present work includes: 1) Prediction of the overall performance map of the research compressor at design and part speed conditions and validation by comparison with experimental results. 2) To obtain the blade element performance of the compressor stage at 90% design speed at choked flow and near stall flow conditions and to compare with experimental results. 3) To investigate the end wall flow fields to understand the flow behaviour of the compressor stage.

TEST FACILITY AND RESEARCH COMPRESSOR

The typical layout of the axial flow compressor research facility of CSIR-NAL is shown in Figure 1. It is an open loop facility consisting of a bell-mouth at the inlet followed by the test compressor stage, volute collector, exhaust ducting, venturimeter and exit throttle valve (ETV). The ETV was used to control the back pressure and hence the mass flow rate of air passing through the compressor. The drive system consisted of 1150 kW DC motor through a speed increasing gearbox (18:1 and 30:1) and an electronic torque meter. Power to the D.C motor is provided by the thyristor unit, which converts the incoming A.C power to D.C power which is then fed to D.C Motor. The thyristor unit is linked with a closed loop speed control system which accurately maintains the set speed of the D. C motor with speed control accuracy of ± 1 RPM. Power to the thyristor unit, is provided by 11 KV/750 V converter transformer [7]. The nucleus of the test facility consists of a single stage transonic compressor stage comprising a NACA transonic rotor with 21 blades and a subsonic stator with 18 vanes. The rotor tip clearance is 1% of the rotor tip axial chord [11].

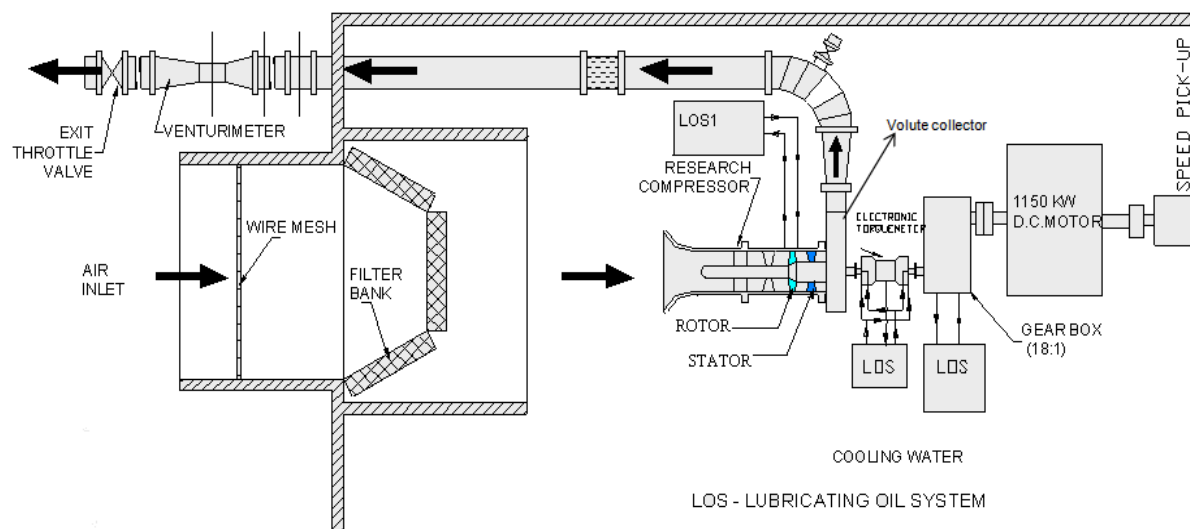


Figure1. SCHEMATIC LAYOUT OF AXIAL FLOW COMPRESSOR RESEARCH FACILITY

Specifications of the compressor

Stage	: Single
Corrected rotational speed	: 12930 rpm
Corrected mass flow rate	: 22 kg/s
Stage total pressure ratio	: 1.35
Stage adiabatic efficiency	: 89%
Rotor tip diameter	: 450 mm

COMPUTATIONAL FLUID DYNAMIC ANALYSIS

3D steady state numerical investigation was carried out using ANSYS FLUENT code at five operating speeds viz. 43%, 60%, 80%, 90% and 100% of the design speed. The overall performance characteristics of the compressor stage were obtained. These values were compared with the available experimental results for validation [12, 13].

Solid modelling of the flow domain

Stage geometry was modelled using commercial CAD tool CATIA. Figure 2 shows the solid model of the compressor stage. Single periodic sector was modelled for the fluid domain and rotational periodic interface was provided.

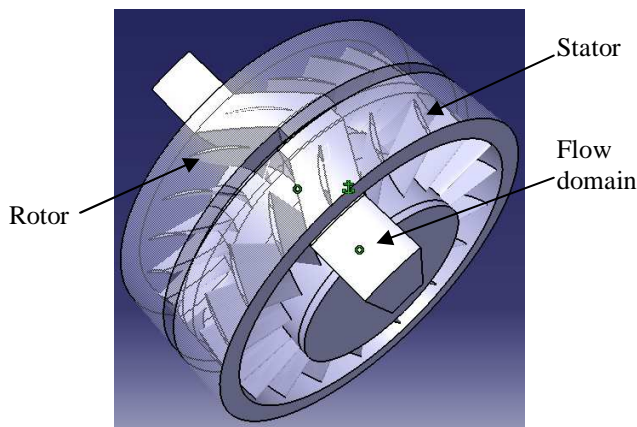
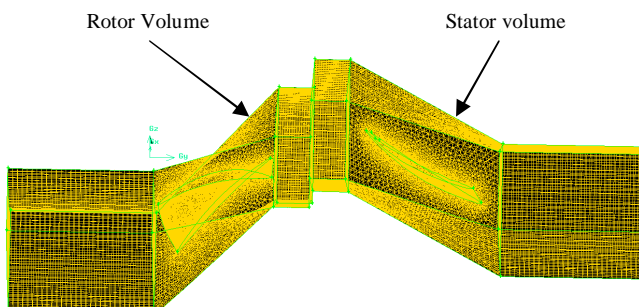


Figure 2. COMPRESSOR STAGE WITH FLOW DOMAIN

Grid generation

The grid was generated for the flow domain using GAMBIT. Size function was used for resolving boundary layers. The flow domain was meshed with hybrid grids as shown in figure 3.



Numerical algorithm:

3-D steady Navier-Stoke equations are solved using segregated pressure based implicit solver of ANSYS FLUENT. Standard k- ϵ turbulence viscosity model with SIMPLE pressure-velocity coupling and First-Order Upwind discretization scheme was used for the analysis.

Boundary conditions

For the rotor fluid, the rotational speed is specified whereas the stator fluid is kept stationary. Periodic condition was applied to both rotor and stator fluid. No-slip condition was given for the flow at the wall boundaries of the blade, hub and casing for both rotor and stator fluid. The static pressure at stator outlet was varied to get the stage performance. Various boundary conditions imposed are shown in Figure 4. The mixing plane was defined at the rotor outlet and the stator inlet.

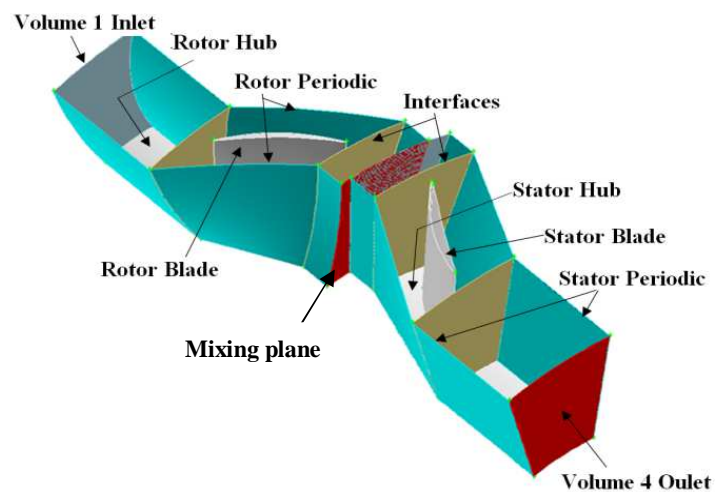


Figure 4. BOUNDARY ZONES SPECIFIED FOR THE FLOW DOMAIN

Grid independence test

In order to ensure that the solution is not affected by the size of the grid, a grid independence study was carried out by taking three different grid configurations and observing their convergence behaviour at 43% of design speed of the compressor. The overall mesh size, worst element skewness and wall y^+ values for the three grid configurations are shown in Table- I. Case-A, Case-B, and Case-C are the three grid configurations. Case-A is the coarser grid with 567020 mesh elements, which resulted in early stall. The Case-B had 1066694 elements, which showed a realistic behaviour and solution was close to experimental stall point. Case-C had 1172597 elements and showed no significant variation in the results as compared to that of Case-B. Consequently, Case-B was chosen for further analysis. The performance characteristics obtained by these three different grid configurations were compared with the experimental results as shown in Figure 5. Corrected mass flow rate is plotted against percentage adiabatic efficiency.

Table 1 Grid parameters

Case	Overall mesh size	Worst element Skewness	Wall y^+ value
A	567020	0.79	50 to 300
B	1066694	0.8	20 to 250
C	1172597	0.8	10 to 200

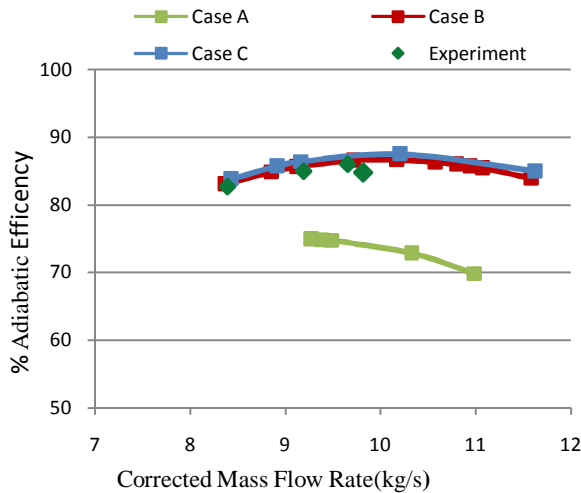


Figure 5. CHARACTERISTIC CURVE OF THE COMPRESSOR OBTAINED FROM DIFFERENT GRID SIZE COMPARED WITH THE EXPERIMENTAL DATA

Solution methodology

The compressor back pressure was varied in steps and the mass flow, pressure ratio and residuals of the system of algebraic equations were used as the convergence criteria to estimate the fluid for every operating condition. In this analysis, energy residual criterion is chosen to be 1×10^{-8} . The characteristics of the compressor were established by determining the mass flow and pressure ratio for each stepwise variation of back pressure. When the compressor was analyzed near stall region, the back pressure variation was maintained in smaller steps. In this region, for each case the back pressure was increased by 500Pa and the characteristics were determined. Stall point was taken as the point after which the solution diverges and fails.

RESULTS AND DISCUSSION

The results obtained from the CFD analysis of the compressor stage are presented as follows. The overall stage performance of the compressor and the radial distribution of flow parameters at the rotor downstream are cited here first to validate the CFD solver, followed by the study of flow field across the compressor using relative velocity vectors and Mach number contours.

Overall performance Map

The predicted overall performance maps obtained for 43%, 60%, 80%, 90% and 100% design speed are plotted and compared with the available experimental results of the test compressor [12, 13]. Figure 6 shows the variation of total pressure ratio and percentage adiabatic efficiency against the corrected mass flow rate. The experimental results are available only for the part speed conditions and hence they are plotted up to 90% design speed. From this figure, it is seen that the CFD results have good agreement with the experimental data at different off-design speeds. The peak efficiency points are marked with black dots on both efficiency plots and the pressure ratio plots. The points labelled as A, B and C on the designed speed efficiency curve represent the different flow conditions viz. choked flow, peak efficiency and near stall flow condition respectively.

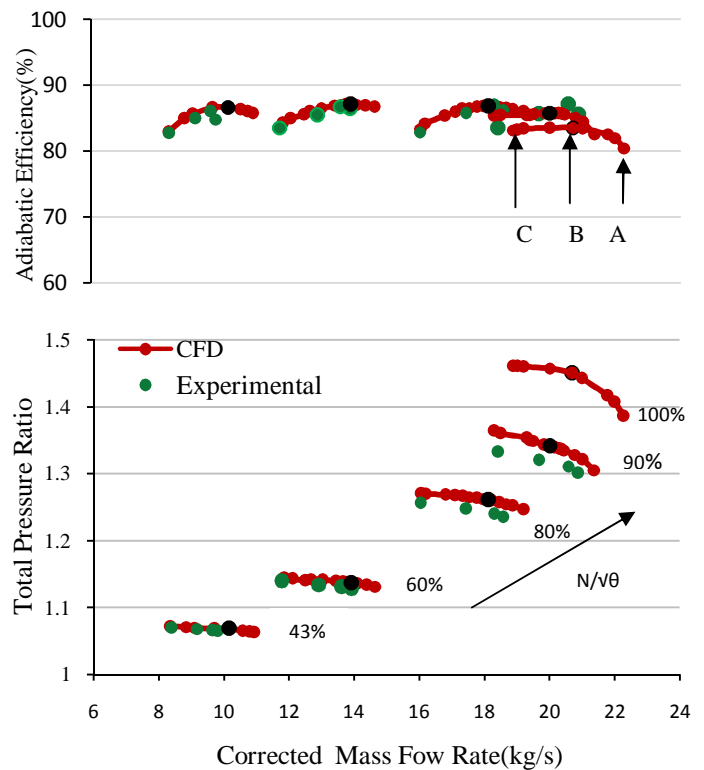


Figure 6. PREDICTED AND EXPERIMENTAL PERFORMANCE MAP OF THE COMPRESSOR

Radial distribution of flow parameter

The predicted radial distribution of flow parameters such as total pressure ratio, static to total pressure ratio and absolute Mach numbers at rotor downstream are compared with the experimental data. Here the flow parameters are presented for two different operating conditions namely at choked flow and near stall flow conditions at 90% design speed. Figure 7 and Figure 8 show the variation of total pressure, static pressure and absolute Mach number along the blade height at choked flow and near stall flow conditions respectively. In these

figures, it is observed that, the radial flow parameters obtained from the CFD analysis are in good concurrence with the experimental results [13].

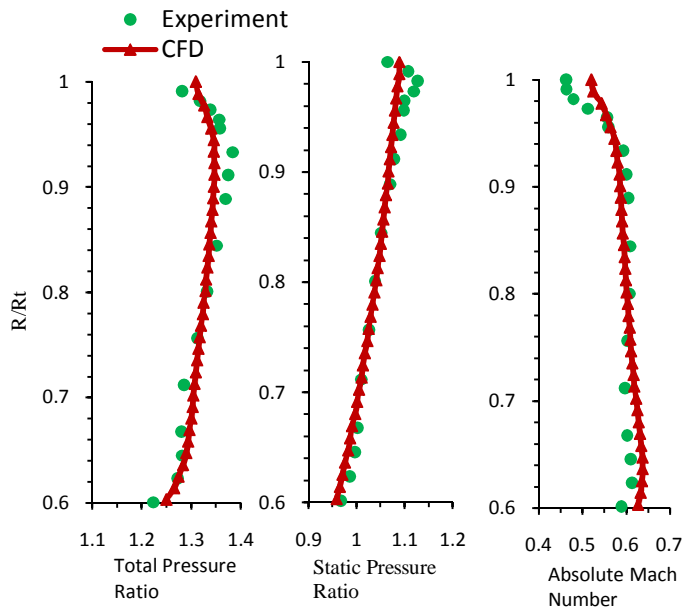


Figure 7. COMPARATIVE RADIAL VARIATIONS OF ROTOR EXIT FLOW PARAMETERS AT CHOKED FLOW CONDITION

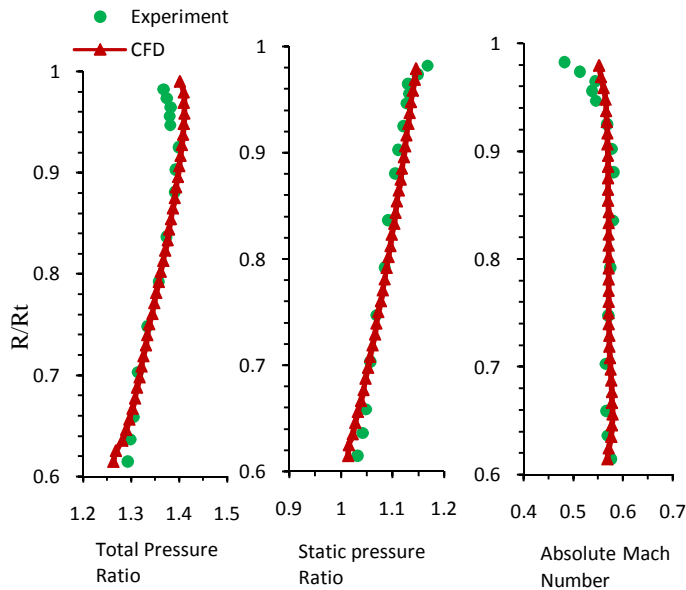


Figure 8. COMPARATIVE RADIAL VARIATIONS OF ROTOR EXIT FLOW PARAMETERS AT NEAR STALL FLOW CONDITION

Flow behaviour of the compressor

Though the computational analyses were carried out for five operating speeds, the computational results are limited to 60% and 100% design speeds. The results are presented in terms of relative velocity vector plots and Mach number contours to identify the key features associated with tip clearance flow and the passage shock structures at these speeds.

Design condition velocity vector plots

Figure 9 shows the relative velocity vectors taken at peak efficiency conditions with three radial spans of the rotor blade to investigate the radial influence of tip leakage flow. Figure 9(a) shows the vectors taken at rotor casing. In this figure, the velocity vectors released in the clearance gap illustrate the formation and trajectory of the leakage vortex. The interaction of tip leakage flow with the main stream flow is clearly visible here. These two flows interface almost at the mid chord of the rotor and formed an angle 'A'. This interaction generated a blockage at the trailing edge of the rotor.

Figure 9(b) shows the relative velocity vectors taken at 94% of rotor span. The radial effect of tip leakage flow and the formation of vortex of low momentum fluid at the suction surface of the rotor at the trailing edge are observed here. This vortex blocked the mainstream flow.

Figure 9(c) shows the relative velocity vectors taken at 85% rotor span at near stall flow condition. This figure shows a smooth flow along the blade surface and the radial influence of tip leakage is negligible.

These numerical results indicate that the key flow features associated with the clearance flow occupy around 15% of the blade outer span at 100% design speed.

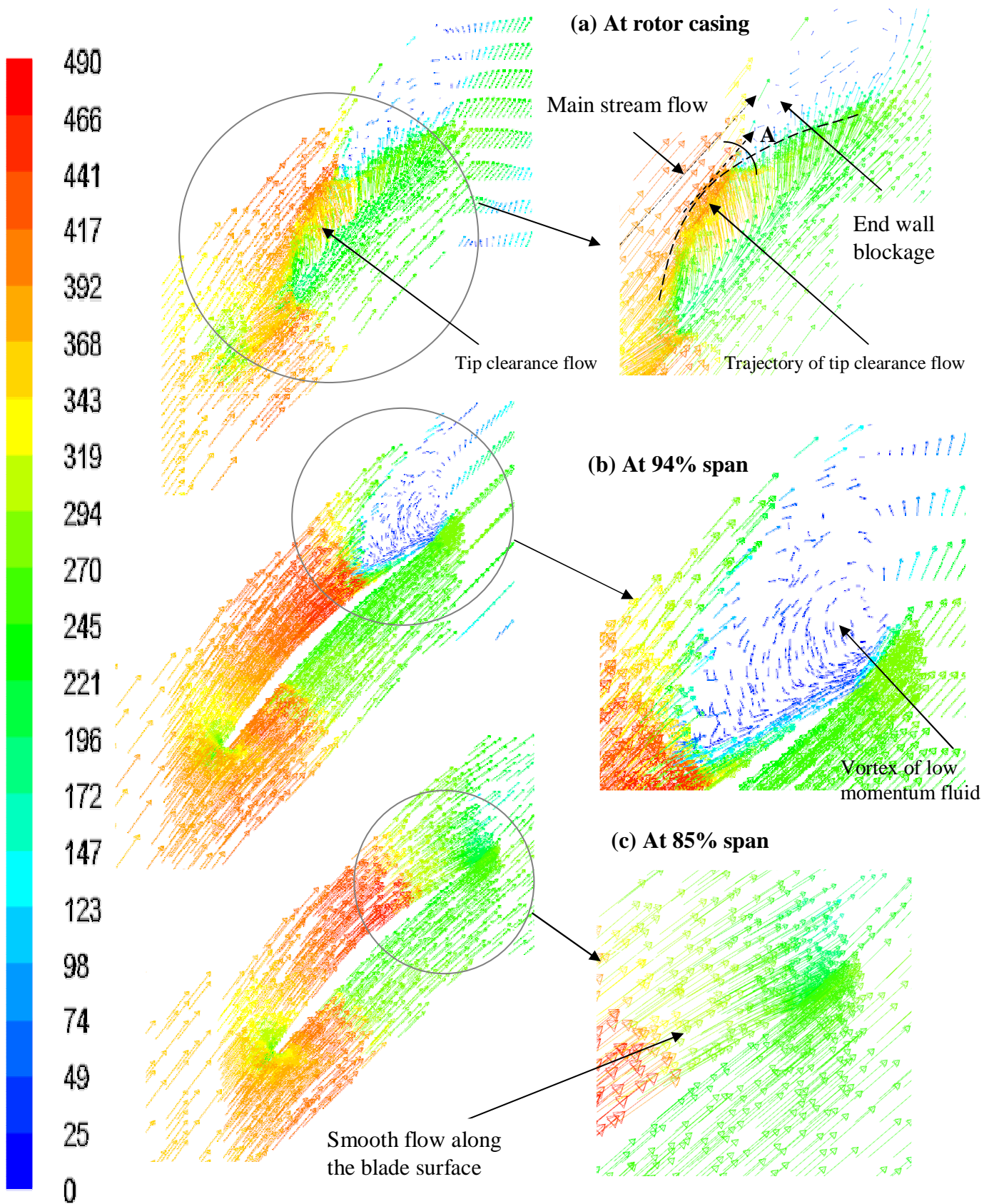


Figure 9. RELATIVE VELOCITY VECTORS AT THREE RADIAL SPAN OF ROTOR AT PEAK EFFICIENCY CONDITION AT 100% DESIGN SPEED.

Part speed velocity vectors

To investigate the radial influence of rotor tip leakage flow in the absence of passage shock, the flow fields are

considered for 60% design speed at peak efficiency and near stall flow conditions.

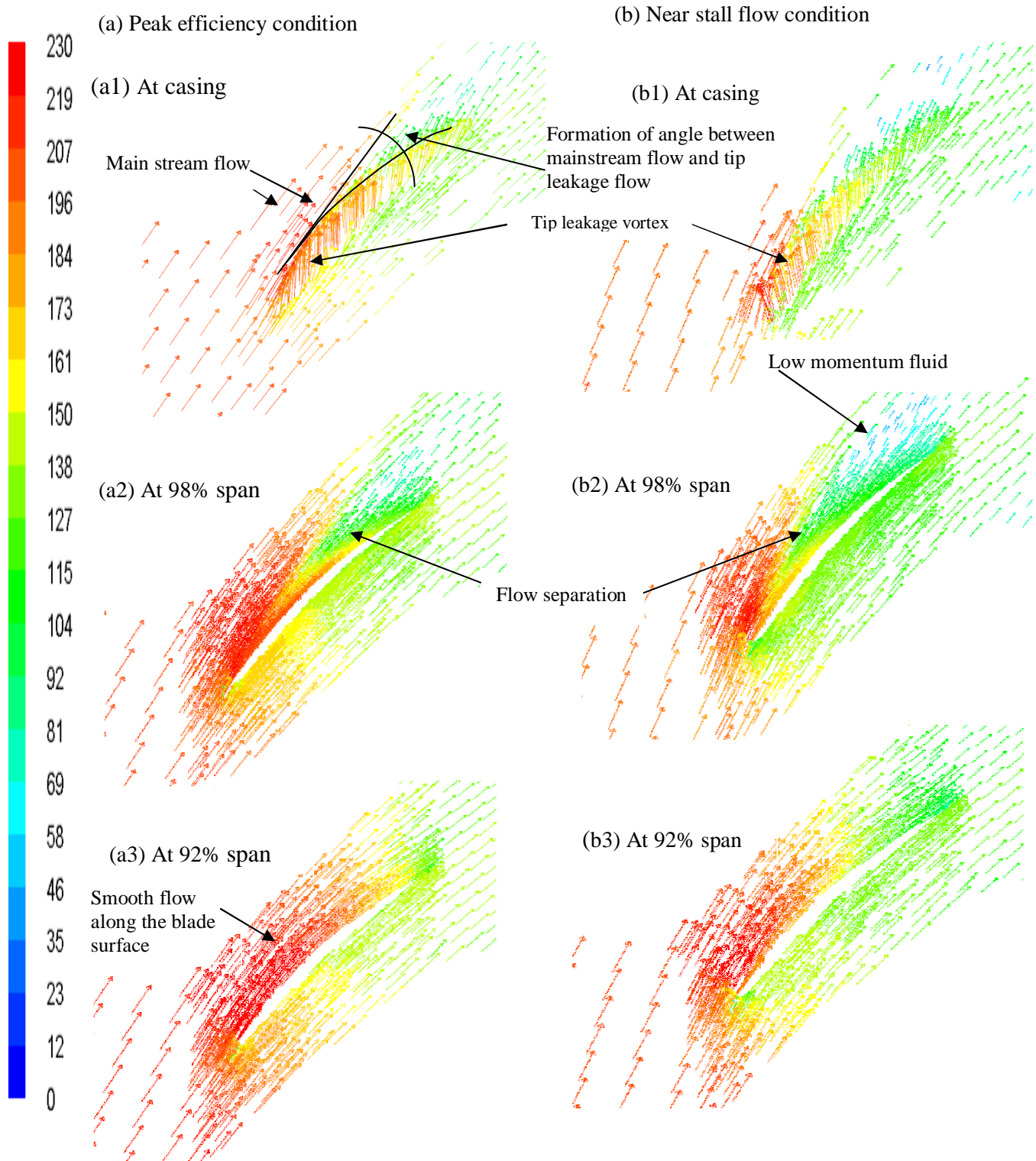


Figure 10. RELATIVE VELOCITY VECTORS AT THREE RADIAL HEIGHT OF ROTOR FOR THE PEAK EFFICIENCY AND NEAR STALL FLOW CONDITIONS AT 60% DESIGN SPEED.

Figure 10 shows the relative velocity vectors taken at three radial spans of the rotor at peak efficiency and near stall flow conditions respectively at 60% design speed condition.

Figure 10(a1) and Figure 10(b1) show the vector plots taken at rotor casing. In these figures, it is observed that the trajectory of the tip leakage flow at 60% design speed is similar to that predicted at 100% design speed. The regions of low momentum fluid indicate the path of the tip leakage vortex and the location of the blockage associated with the leakage vortex.

Figure 10(a2) and Figure 10(b2) show the vector plots at 98% of rotor radial span. In these figures, the flow separation from the suction surface of the blade is clearly visible and this flow separation causes blockage in the mainstream flow. The flow blockage is increased with increasing the compressor loading. No recirculation of low momentum fluid is seen at the trailing edge of the rotor, which was observed in the design speed case.

Figure 10(a3) and Figure 10(b3) shows the vector plots taken at 92% of rotor radial span. In these figures, it is observed that the affect of tip leakage vortex is almost negligible in the case of peak efficiency condition and there are small traces of tip leakage at near stall flow condition. These indicate that the radial influence of the tip leakage vortex is only up to 8% of the blade span at 60% design speed.

Mach number contour plots at design speed

At design speed, comparison of blade to blade relative Mach number (M_{rel}) distribution along the rotor blade surface is discussed at peak efficiency and near stall flow conditions. Mach number contours are taken at two radial locations such as 98% and 94% radial span of the rotor. The Mach number contours are drawn with the same colour bar for both peak efficiency and near stall flow conditions.

Figure 11(a1) and figure 11(a2) show the Mach number contours taken at 98% and 94% span respectively for peak efficiency condition. Figure 11(b1) and figure 11(b2) are taken at near stall flow condition. In these figures, it is seen that at peak efficiency, the passage shock is oblique and it is attached to the leading edge at pressure surface, while at near stall the shock is detached from the leading edge. At both operating conditions, a region of low Mach number fluid originates near mid-pitch just downstream of the shock and migrates toward the pressure surface of the blade. At the peak efficiency condition, the fluid in this region merges with the wake downstream of the blade. Similar observations are quoted in [3].

Mach number contour plots at part speed

Analogous to the design speed case, computational results are presented for 60% design speed to identify key features associated with passage shock. The computations at

60% design speed utilised the same blade geometry as that used at design speed. Relative Mach number distribution on the rotor blade is discussed at peak efficiency and near stall flow conditions at 98% and 94% blade spans. The Mach number contours are drawn with the same colour bar for both operating conditions. Figure 12(a) and figure 12(b) show the relative Mach number contours taken at peak efficiency and near stall flow conditions respectively. The regions of low Mach number indicate the path of the tip leakage vortex and the location of the blockage associated with the vortex. These figures also signify the flow blockage is increased with increasing the compressor loading in both radial as well as circumferential direction.

CONCLUSIONS

Steady state computational analyses were carried out to validate the flow solver and to investigate the flow behaviour of the single stage transonic axial compressor. Performance characteristics of the compressor obtained at design and various part speed conditions. The radial distribution of flow parameters like total pressure, static pressure and absolute Mach numbers are also obtained from the numerical calculation at 90% design speed. The predicted results are validated with the experimental data. The flow field of the compressor are studied with the help of relative velocity vectors and Mach number contour plots.

The results of the investigation are:

1. The implemented CFD flow solver ANSYS FLUENT yielded a result which is close to the experimental values of the test rig.
2. Visualized a strong interaction between the rotor passage shock and the tip leakage vortex, which generated a region of high blockage in the blade passage at design speed condition.
3. The blockage moves upstream of the rotor and becomes larger in both circumferential and radial directions as the rotor loading is increased.
4. The influence of the shock and the tip leakage vortex interaction extends around 15% of the rotor outer span at design speed condition.
5. The radial influence of tip leakage flow within the blade is around 8% of the rotor outer span.

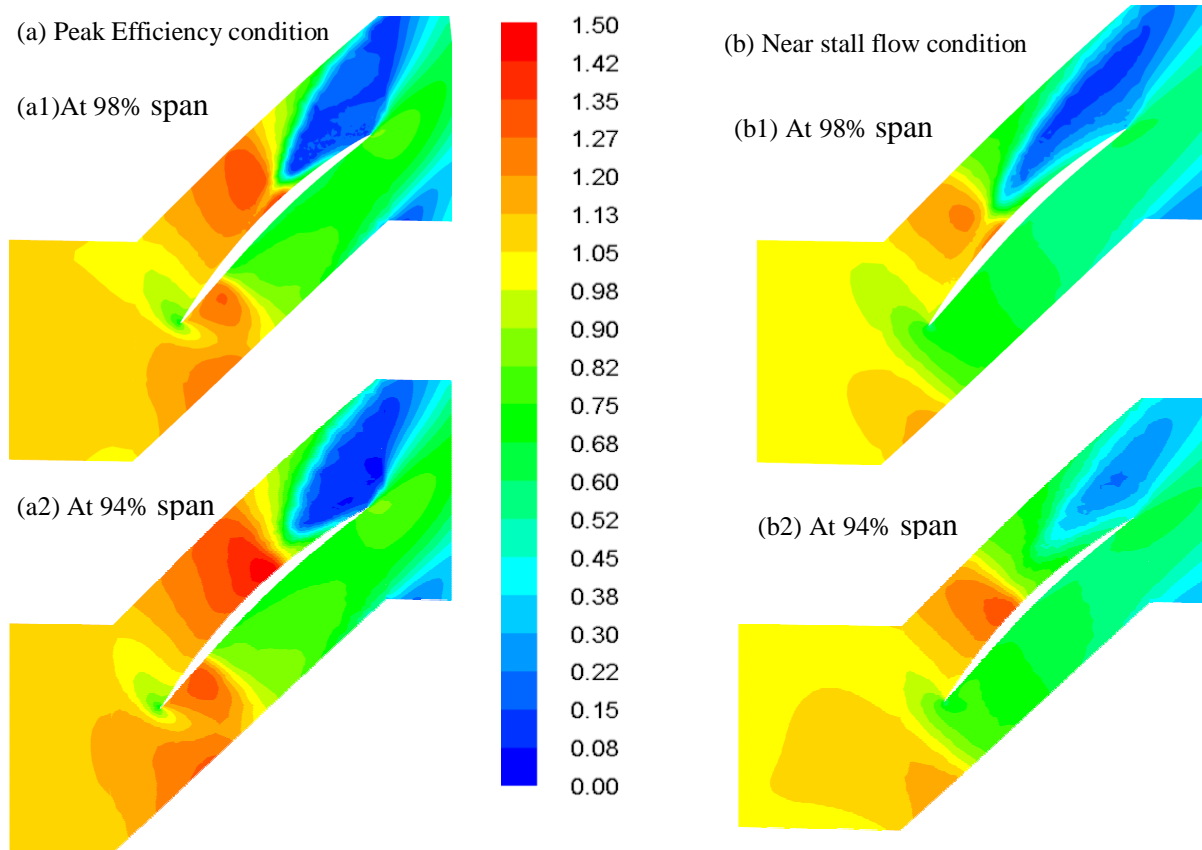


Figure11. RELATIVE MACH NUMBER CONTOUR PLOTS AT (a) PEAK EFFICIENCY AND (b) NEAR STALL FLOW CONDITION AT 100% DESIGN SPEED.

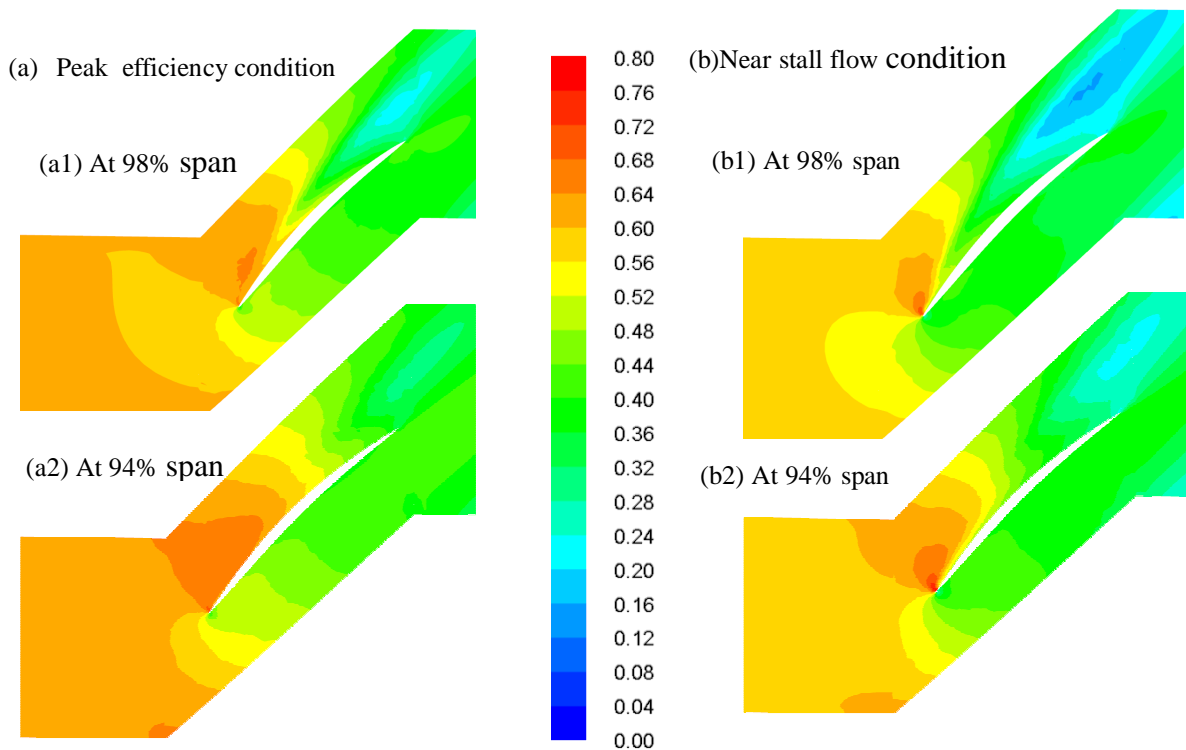


Figure12. RELATIVE MACH NUMBER CONTOUR PLOTS AT (a) PEAK EFFICIENCY AND (b) NEAR STALL FLOW CONDITION AT 60% DESIGN SPEED

ACKNOWLEDGEMENTS

The authors would like to thank Director, CSIR-NAL for permitting this study at NAL. Sincere thanks goes to Head, Propulsion Division for all the support. The thanks are also due to all the members of Axial Flow Compressor Rig for their cooperation during execution of this work.

REFERENCES

- [1] Robert Biollo and Ernesto Benini, "State-of- Art of Transonic Axial Compressors", *University of Padova, Italy*.
- [2] Jennions I.K and Turner M G, April 1993, " Three-Dimensional Navier -Stokes Computations of Transonic Fan Flow Using an Explicit Flow Solver and an Implicit k- ϵ Solver. *ASME Journal of Turbomachinery, Vol.115/265*,.
- [3] Kenneth L Suder and Celestina, M L., April 1996, "Experimental and Computational Investigation of The Tip Clearance Flow in a Transonic Axial Compressor Rotor", *ASME Journal of Turbomachinery, Vol.118, No2, , pp 218-229, ISSN 0889-504X*.
- [4] Anton Weber et al, July 2002, "3-D Transonic Flow in a Compressor Cascade with Shock-Induced Corner Stall". *ASME Journal of Turbomachiner. Vol. 124*.
- [5] Denton & Xu, 1999, "The exploitation of Three Dimensional Flow in Turbomachinery Design", *Journal of Mechanical Engineering Science, Vol 213, No.2 1999*.
- [6] Chima, R V , January 1998, "Calculation of Tip Clearance Effects in a Transonic Compressor Rotor", *ASME Journal of Turbomachinery, Vol 120, No.1. Pp131-140, ISSN 0889-504X*.
- [7] Adamczyk J J, Celestina M.L, and Greitzer E M, 1993, " The role of Tip Clearance in High Speed Fan Stall", *ASME Journal of Turbomachinery, Vol. 115, pp 28-38*.
- [8] Copenhaver W W, Hah C and Puterbaugh S L, 1992,"Three- Dimensional Flow Phenomena in a Transonic, High Through-Flow, Axial Flow Compressor stage" *ASME Paper No. 92-GT-169*.
- [9] Storer J A and Cumpsty N A, 1993,"An Approximate Analysis and Prediction Method for Tip Clearance Loss in Axial Compressors," *ASME 93-GT-140*.
- [10] Koch C and Smith L 1976, " Loss sources and Magnitudes in Axial Flow Compressors", *ASME Journal of Engineering for Power, Vol. 98, pp 411-424*.
- [11] Mohan K, .Nagpurwala Q H, Guruprasad S A, 1981, "High speed Axial Flow Compressor Research Facility", *NAL-TM-PR- UN-0-(103), 103/1/1981*.
- [12] Guruprasad S.A, 1999, "Experimental Investigations on the Influence of axial extension and Location of Outer Casing Treatment on the Performance of an Axial Flow compressor" *4th Intersym AIF, Dresden Germany*.
- [13] Guruprasad S.A. and Shobhavathy M T, Jan 10-11, 2003, "Flow behavior in an Axial flow Compressor having Casing Treatment", *VI National Conference on Air Breathing Engines, Bangalore*.
- [14] Hanoca and Shobhavathy M T, July 2011, "A CFD Analysis to Study the Influence of Axial Spacing on the Performance of a Single Stage Transonic Axial Flow Compressor", *Propulsion Division, M Tech Project Dissertation*.